ПРОЕКТСОФТ

Software Package

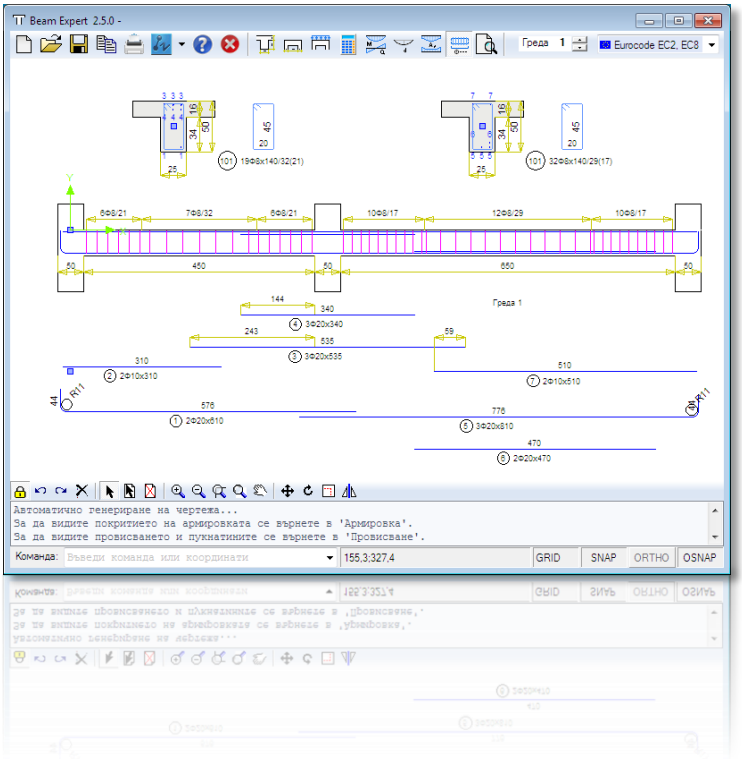
Design Expert version 2.7

Structural Design and Detailing to Eurocode

Beam Expert

Analysis, design and detailing of RC continuous beams

User manual



2014

TABLE OF CONTENTS

About the program

Beam expert is a software product for static analysis, design, detailing and drafting of reinforced concrete continuous beams according to Eurocode (EC2, EC8 etc.). It is part of Design Expert software package. Main features of the program are:

Static analysis

Dimensions of cross sections, spans, columns and support types are entered in tables. Dead, live and seismic loads should be defined as well. Static analysis are performed for different load patterns that account for the most unfavorable load position. Envelope diagrams of internal forces and elastic deformations are obtained as a result. Values at column edges are calculated as well.

Additionally, inelastic deflections, crack width and capacity values of internal forces can be calculated according to Eurocode. Reinforcement counts and diameters should specified before that.

The program generates a detailed calculation report including input data and results. The report is saved in HTML format for printing and viewing.

Design

Bending and shear design checks are performed for multiple sections along all spans. Diagrams for main and shear reinforcement are obtained as a result. Additional tensile force in main reinforcement due to shear load is taken into account.

Detailing

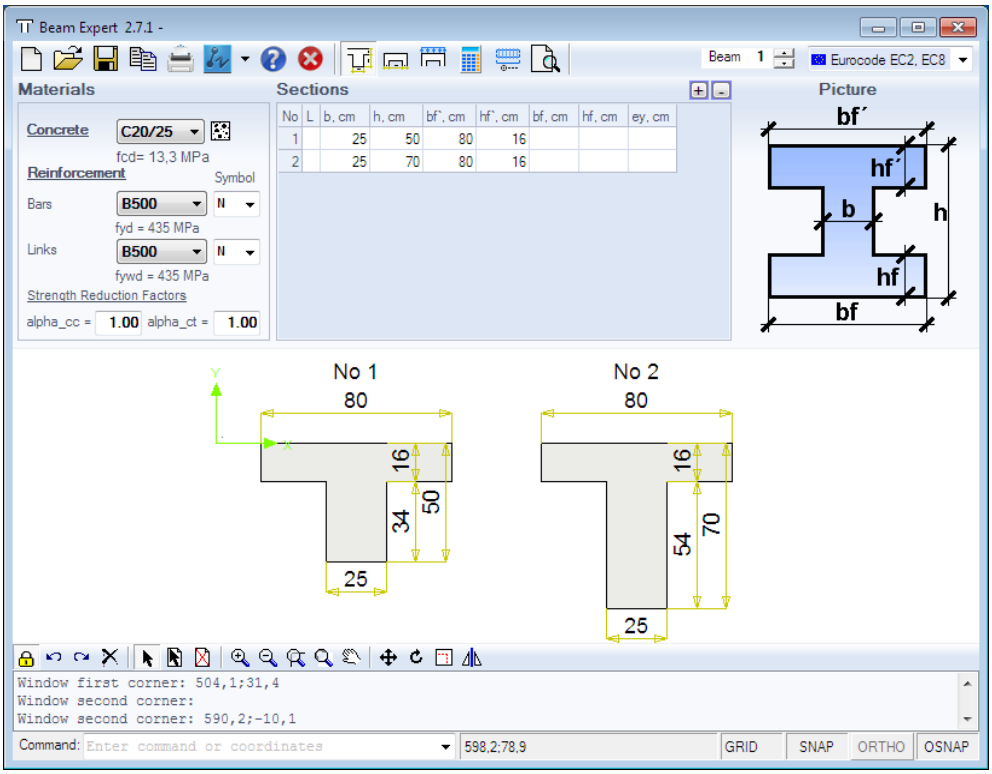
Bar counts, diameters and lengths are automatically determined from the required reinforcement diagrams. Detailed reinforcement drawings with views and sections are generated. Most of Eurocode detailing provisions like reinforcement ratios, anchoring, splice lengths, bar and link spacing, critical zone lengths etc. are included into the program. They are applied automatically during the detailing process. The generated reinforcement can be modified and edited by the user at each stage.

Drafting

Drawing is created firstly in Design Expert internal graphical editor where you can review and modify. Then you can export it directly to ZWCAD+ or AutoCAD or save a script file for AutoCAD LT. The software generates bills of materials for both steel and concrete and reinforcement bending schedule. The reinforcement output is compatible to the Design Expert Plug-in module for reinforcement detailing and scheduling with AutoCAD or ZWCAD+.

How it works?

The software includes standard graphical user interface for Windows. You can enter commands either by clicking buttons on the main toolbar or by typing commands in the command line located at bottom. Detailed descriptions of all commands are provided further in this manual. If you hold the mouse over a button, a tooltip appears with a short description of the respective command.



Working with files

Beam Expert has its own file format which is used to save program data permanently on the disk. Input file extension is ***.bea**. Results are stored into ***.bea.html** files.

New file

Click the button to save current data to a new file. A standard file selection dialog appears on screen. Select or write down file path and name and click **"Save".**

Open a file

Click the button to open a file from the disk. A standard file selection dialog appears on screen. Browse for the file using the mouse or type file path and name and click **"Open".**

Save a file

Click the button to save a file to the disk. A standard file selection dialog appears on screen. Select the destination folder and file name. If file already exists you will be prompted to overwrite or change the name.

Input data

Input data is divided into several pages for convenience. You can switch between pages by clicking the respective buttons on the main toolbar  . Use the text fields and tables inside each page to enter data. The results are shown immediately on the drawing at the bottom.

Working with tables

Most of the input data is entered in tables. You can use the following commands to work with all tables inside the program:

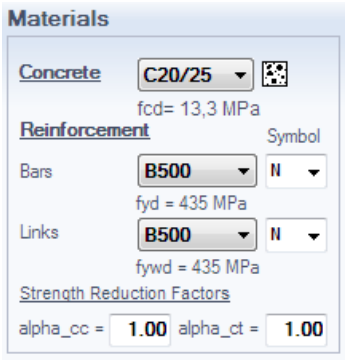
- Insert new row – press **Ins** key or "**+**" button. When you go to the end of the last row and press **Enter**, a new row opens automatically;
- Delete last row – press **Backspace** or "**–**" button. Some tables are with fixed dimensions and you cannot add or remove rows;
- Move the current focus with one cell – press arrow keys ←, ↑, ↓, →;
- Move the focus to the first or the last row – press **Page Up**, **Page Down**, **Home**, **End**;
- Edit current cell contents – press **F2** or just start typing – an input box is opened automatically;
- Finish cell edit – press **Enter** or arrow key – the input box is closed and changes are stored into the cell;
- Cancel cell edit – press **Esc** – the input box is closed and changes are discarded. The original contents remains in the current cell;
- Delete cell contents – press **Del** – the contents of all selected cells is cleared;
- Select a range of cells – the first method is to use the keyboard – select the first cell, hold **Shift** and press arrow keys or **Page Up**, **Page Down**, **Home**, **End** to move to the cell at the other corner of the area. Alternatively, you can click with the mouse at the first corner, hold **Shift** and click at the opposite corner;
- Copy the contents of the selected cells – press Ctrl+C;
- Paste into the selected cells – press Ctrl+V;

You can copy from and paste to the same or other tables as well as external programs like Word, Excel, etc. If you try to paste a range of cells which area is greater than the area of the destination cells, you will receive a warning. This is necessary to avoid unwanted data overwriting.

Design code

Design Expert is compatible to Eurocodes, mainly EN 1992-1-1 and EN 1998-1-1. It is applicable to most countries as far as you can define your own material properties, partial safety factors, loads and some other important parameters. Detailed description of all design methods and formulas used in this program is provided further in this manual.

Materials



Material and section properties are entered on the first page of the program. It is active at startup by default. If you have moved to another page, you can always go back with the button.

You have to enter concrete grade C20/25 and steel grades for main B500 and shear B500 reinforcement. Also, you can specify a symbol (N, Ø, ect.) to be used for bar labelling in the drawings. Characteristic and design values for material properties are predefined in tables. Concrete compressive and tensile strengths are additionally multiplied by the sustained loading factors α_{cc} and α_{ct}. They should be defined separately in the respective fields since they are not included in the table values.

Material tables

You can open the material tables by clicking the button. A dialog containing both concrete and reinforcement tables appears on screen. You can modify values, add and remove rows by clicking the „**+**" and „**-**" buttons, respectively. Finally you should press "**Save**" to save changes and close the dialog. If you want to discard changes, press "**Exit**" and you will return to the main window.

Material tables are common for the whole computer. Any changes you make will reflect all Design Expert modules and input files.

Material data to Eurocode

Concrete

Design Expert includes the following concrete grades according to EN 1992-1-1, Table 3.1:

Name	E_{cm} GPa	$f_{ck,cube}$ MPa	f_{cd} MPa	f_{ctd} MPa	f_{ck} MPa	$f_{ctk,0.05}$ MPa	ε_{c2}	ε_{cu2}
C12/15	27.0	15.00	8.00	0.73	12.00	1.10	0.002	0.0035
C16/20	29.0	20.00	10.67	0.87	16.00	1.30	0.002	0.0035
C20/25	30.0	25.00	13.33	1.00	20.00	1.50	0.002	0.0035
C25/30	31.5	30.00	16.67	1.20	25.00	1.80	0.002	0.0035
C30/37	33.0	37.00	20.33	1.33	30.50	2.00	0.002	0.0035
C35/45	34.0	45.00	23.33	1.47	35.00	2.20	0.002	0.0035
C40/50	35.0	50.00	26.67	1.67	40.00	2.50	0.002	0.0035
C45/55	36.0	55.00	30.00	1.80	45.00	2.70	0.002	0.0035
C50/60	37.0	60.00	33.67	1.93	50.50	2.90	0.002	0.0035

The following symbols are used in the above table:

E_{cm} – concrete secant modulus of elasticity;

$f_{ck,cube}$ – characteristic cube strength;

f_{ck} – characteristic cylinder strength;

$f_{ctk,0.05}$ – characteristic tensile strength with 5% probability of failure;

$f_{cd} = \alpha_{cc} f_{ck}/\gamma_c$ – design compressive strength;

$f_{ctd} = \alpha_{ct} f_{ctk,0.05}/\gamma_c$ – design tensile strength;

ε_{c2} – compressive strain at maximum stress for parabolic-linear stress-strain;

ε_{cu2} – ultimate compressive strain at concrete edge.

Design values for compressive and tensile strengths in the table are determined for partial safety factor γ_c = 1.5. They still do not include α_{cc} and α_{ct} factors which should be defined additionally. Some countries use α_{cc} = 0.85 and α_{ct} is usually equal to 1.0. You should look for these values in your national annex document.

Reinforcement

Design Expert includes the following reinforcement steel grades:

Name	E_s GPa	f_{yd} MPa	f_{yk} MPa	ε_{yd}
B220	200	191	220	0.01
B250	200	217	250	0.01
B420	200	365	420	0.01
B460	200	400	460	0.01
B500	200	435	500	0.01

The following symbols are used in the table:

E_s – design modulus of elasticity;

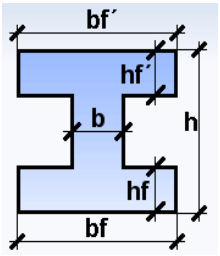
f_{yd} – design yield strength;

f_{yk} – characteristic yield strength;

ε_{yd} – design ultimate strain.

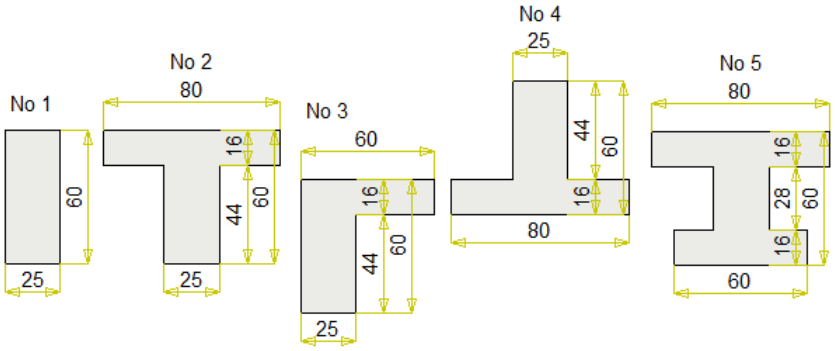
Cross sections

Cross section data is entered in the "**Sections**" table. Shapes can be "T", inverted "T", "I", "L" or inverted "L". Dimensions are noted as shown in the picture on the right. For „L" or inverted "L" shapes enter „L" in the first column. Sections are aligned to their top edges by default. You can move a section up or down by entering, respectively, positive or negative value for the eccentricity „**ey, cm**". That is how, you can align sections to their bottom edges, in case of beam with slab at bottom. Sample input data for different types of cross sections is provided in the table below:

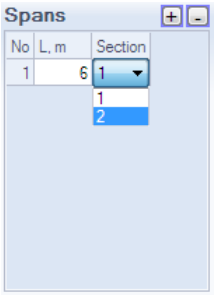


No	L	b, cm	h, cm	bf`, cm	hf`, cm	bf, cm	hf, cm	ey, cm
1		25	60					
2		25	60	80	16			
3	L	25	60	60	16			-22
4		25	60			80	16	22
5		25	60	80	16	60	16	22

Pictures of all sections in the table and their dimensions are displayed in the bottom half of the main window. The result from the above input will look as follows:

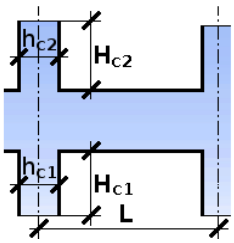


Spans



Span data is entered in the "**Spans**" table on the respective page. Click the "**Spans**" button to activate it. Add as many rows as you need. For each span, fill the effective length (distance between centers of columns) and select the number of the cross section. It must corresponds to a row in the "**Sections**" table.

Supports

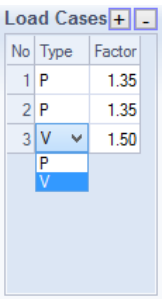


Enter support type and column dimensions above and below the beam at each support. Types of supports are ***F*** – fixed, ***P*** – pinned and ***C*** – cantilever (free end). Column size "h_{ci}" is along the beam and "b_{ci}" is across the beam. If you enter positive value for column height "H_{ci}", the respective column stiffness will be included in the analysis. The beam will be calculated as a part of frame. In this case, the defined support types refer to column ends. Index i = 1 refers to the lower column and index i = 2 - to the upper one.

Supports And Column Sizes, cm

No	Type	hc1, cm	bc1, cm	Hc1, cm	hc2, cm	bc2, cm	Hc2, cm
1	P		25			25	
2	P		25			25	
3	P		25			25	

Loads



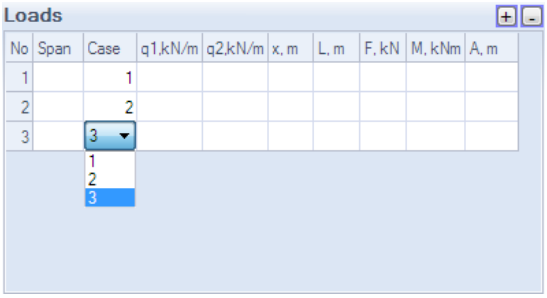
Press the "**Loads**" button to open the respective page. First you have to define all load cases. Add as many rows as you need in the "**Load Cases**" table. For each load case, select case type and enter partial safety factor. Load types can be ***P*** for permanent and ***V*** for variable. Characteristic values should be entered for all loads in Beam Expert. Then ULS combination will be automatically generated and loads will be multiplied by the defined safety factors. Partial safety factors have the following values by default:

- permanent loads: $\gamma_G = 1.35$
- variable loads: $\gamma_Q = 1.50$

In case of more than one variable loads, they must be multiplied by additional combination factors ψ. Design load combination for ULS is defined by the following equation, according to Eurocode:

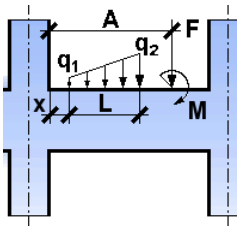
$$\Sigma\gamma_G \cdot G_{kj} \text{ "+" } \gamma_{Q,1}\, Q_{k,1} \text{ "+" } \Sigma\gamma_{Q,i} \cdot \psi_{0,i}\, Q_{k,i};$$

Variable load with major contribution to the internal forces is assumed to be leading "$Q_{k,1}$". It is multiplied by partial safety factor "$\gamma_{Q,1}$" only. All remaining loads should be multiplied by $\gamma_{Q,i} \cdot \psi_{0,i}$. Values should be entered in the "**Factor**" column in the table. You can find load combination factors in your national Annex.



Load values for each span and case are entered in the „**Loads**" table. Loads can be distributed or concentrated forces and moments.

You can add as many rows as you need by clicking the "+" button. Enter span number in the first column. If you leave it empty, the load will be applied to all spans. If the load should act on several spans, you can enter their numbers, separated by commas (e.g. "1,2,4,7"). Enter load case number in the second column. Double click to open a drop down list with possible values. You can have many loads in one load case.



Distributed loads can be uniform or linear. You have to enter left and right values q_1 and q_2 as shown in the picture. When you fill q_1 and press Enter, the value is copied automatically into q_2. That helps you to enter uniform loads more quickly. For linear loads, set different value of q_2. Loads can be partially distributed along spans. In this case, you have to define additionally:

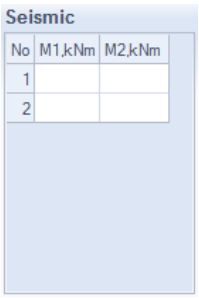
x – starting distance from the edge of the left column;

L – length of load distribution.

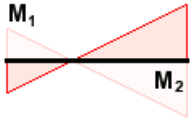
If you leave x and L empty, the load will be distributed along the full span clear length. If you enter $x > 0$ and $L = 0$, then load will start at x and continue to the edge of the right column. It is the same if you enter L to be greater than the remaining of the span.

Concentrated forces and moments are defined by load values „**F**" and „**M**" and distance „**A**" from the left column edge. You can enter distributed and concentrated loads in the same row as long as they act in the same span and in the same load case.

Loads can be located only within clear span length and not within columns.



If the beam is part of an anti-seismic frame structure, you can enter seismic moments M_1 and M_2 at both ends of each span. They can be obtained separately by seismic analysis software using finite element model. Values should be calculated at column centers. Signs are neglected and only absolute values for M_1 and M_2 are important. The program automatically creates two combinations that correspond to both directions of seismic loading: right ($-M_1$, $+M_2$) and left ($+M_1$, $-M_2$). Finally it calculates envelope diagrams, including permanent and variable loads in seismic design situation. That is why, you have to define the ψ_2 combination factor for variable loads. Permanent and seismic load combination factor is always 1.00.



Results

Analysis

Only buttons for input pages are shown in the main toolbar before running the analysis. When you finish all the input, press the button to start the analysis. Buttons for output pages are loaded and internal forces

diagrams are displayed on screen. You can always go back to the diagrams by pressing the "**Diagrams**" button.

Elastic analysis of the beam is performed by using matrix method (direct stiffness method). Each span is divided into 12 segments. Element stiffness is calculated using cross section properties and mean secant modulus of elasticity of concrete E_{cm}. Supports are located at column centers and rigid end zones are added between column edges.

The program automatically generates three load combinations according to Eurocode:

ULS (Ultimate Limit State) - $\Sigma\gamma_G \cdot G_{kj}$ "+" $\gamma_{Q,1} \cdot Q_{k,1}$ "+" $\Sigma\gamma_{Q,i} \cdot \psi_{0,i} \cdot Q_{k,i}$;

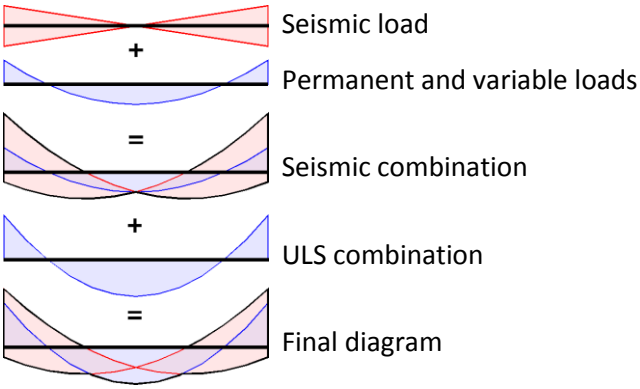
SLS (Serviceability Limit State) - ΣG_{kj} "+" $Q_{k,1}$ "+" $\Sigma\psi_{0,i} \cdot Q_{k,i}$;

Seismic - ΣG_{kj} "+" $\Sigma\psi_{2,i} \cdot Q_{k,i}$ "+" A_{Ed}.

Internal forces are calculated using the ULS and seismic combinations while deformations and crack widths are calculated by the SLS combination.

Internal forces

Bending moment diagram M and shear force diagram V are obtained as a result. The most unfavorable distribution of variable loads is taken into account and envelope diagrams are created. If you have seismic loads, the final diagram is created by enveloping ULS and seismic combinations.



Capacity design

You can use Beam Expert to obtain the capacity values of internal forces and include them into the reinforcement design. Before that, you have to design and draw the reinforcement using the initial values as a first iteration. Click the "**Reinforcement**" and "**Drawing**" buttons to design and draw the reinforcement and select bar diameters and counts. Then go back to internal forces diagrams by clicking the "**Diagrams**" button and you will see the capacity moments and shear forces. They are labeled right below and above beam axis at both ends. They are calculated only if medium (DCM) or high (DCH) ductility class is selected.

Capacity bending moments

Capacity values of bending moments $M_{Rd,t}$ and $M_{Rd,b}$ are calculated for top and bottom reinforcement, respectively, at both ends of each span. They represent the maximum bending moments that a cross section can take with the selected reinforcement. In general case of a Tee section, they are calculated as follows:

$M_{Rd} = F_s \cdot z$, where

$F_s = f_{yd} \cdot A_{s,prov}$ – tensile force in main reinforcement;

$z = d - 0.5(A_{bf} \cdot h_f + b \cdot x^2)/(b \cdot \lambda x + A_{bf})$ – lever arm of internal forces;

$\lambda x = (F_s/f_{cd} - A_{bf})/b$ – compression zone depth;

Values of A_{bf} and b are determined as follows:

Rectangular cross section	Tee cross section	
	If $F_s > b_f \cdot h_f \cdot f_{cd}$ neutral axis in web	If $F_s \le b_f \cdot h_f \cdot f_{cd}$ neutral axis in flange
$A_{bf} = 0$; $b = b_w$	$A_{bf} = (b_f - b_w) \cdot h_f$; $b = b_w$	$A_{bf} = 0$; $b = b_f$

b_w – web width;

b_f – width of compressed flange;

h_f – thickness of compressed flange;

$d = h$ – 5 cm – effective depth;

h – section height;

f_{yd} – design yield stress;

f_{cd} – design compressive strength of concrete;

$A_{s,prov}$ – area of reinforcement at the respective side (top or bottom), that is sufficiently anchored behind column edge.

Soft storey check

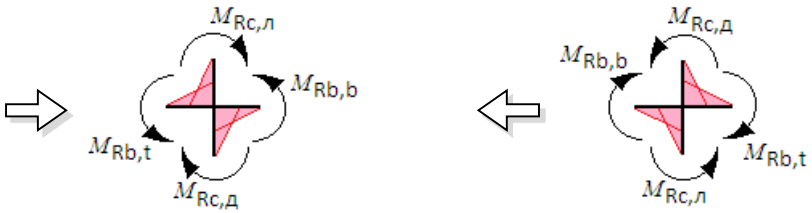
According to Eurocode 8, columns in anti-seismic frame structures should be designed with greater bending capacities than the adjacent beams. Capacities are determined so that the following condition to be satisfied in each beam-column joint:

$\sum M_{Rc} \ge 1.3 \sum M_{Rb}$, where

$\sum M_{Rc}$ – sum of bending capacities of columns above and below the joint;

$\sum M_{Rb}$ – sum of bending capacities of all beams connected to the joint in the selected plane.

Design check must be performed for both directions of seismic loading. Capacity moments at the respective sides should be taken for each direction:



Capacity moments for beams $M_{Rd,t}$ and $M_{Rd,b}$ can be calculated by Beam Expert (see the above chapter). Capacity moments for columns can be calculated using PMM Expert, which is another Design Expert module. For convenience, you can export diagrams with capacity values to AutoCAD or ZWCAD by using the  or  buttons.

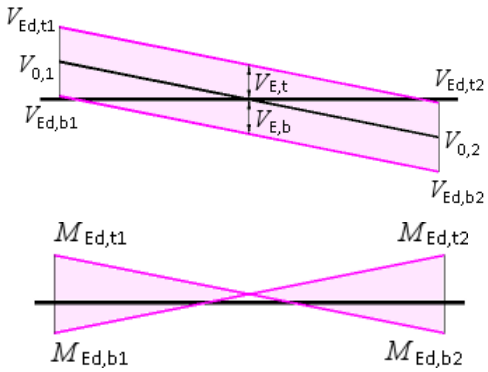
Capacity shear forces

According to Eurocode 8 design shear forces have to be determined using bending capacity moments at ends of each span. Capacity shear force diagrams are generated. Values are displayed above and below beam axis at both ends. They are calculated as follows:

at the left end (1): at the right end (2):

$V_{Ed,t1} = V_{0,1} + V_{E,t}$ $V_{Ed,t2} = V_{0,2} + V_{E,t}$

$V_{Ed,b1} = V_{0,1} + V_{E,b}$ $V_{Ed,b2} = V_{0,2} + V_{E,b}$



where $V_{0,1}$ and $V_{0,2}$ are design shear forces due to vertical loads in seismic load combination:
$\sum G_{kj}$ "+" $\sum \psi_{2,i} \cdot Q_{k,i}$

$V_{E,t} = (M_{1d,t} + M_{2d,b})/l_{cl}$

$V_{E,b} = (M_{1d,b} + M_{2d,t})/l_{cl}$

Moments $M_{1d,t}$, $M_{2d,t}$, $M_{1d,b}$ and $M_{2d,b}$ are the increased values of the capacity moments at both ends. They are determined by the following equations:

$M_{1d,t} = \gamma_{Rd} \cdot M_{Rd,t1} \cdot \min(1; \sum M_{Rc,1}/\sum M_{Rb,1})$ $M_{2d,t} = \gamma_{Rd} \cdot M_{Rd,t2} \cdot \min(1; \sum M_{Rc,2}/\sum M_{Rb2})$

$M_{1d,b} = \gamma_{Rd} \cdot M_{Rd,b1} \cdot \min(1; \sum M_{Rc,1}/\sum M_{Rb,1})$ $M_{2d,b} = \gamma_{Rd} \cdot M_{Rd,b2} \cdot \min(1; \sum M_{Rc,2}/\sum M_{Rb2})$

According to Eurocode, the factor γ_{Rd} is assumed to be 1.0 for DCM and 1.2 for DCH. Capacity moments $M_{Rd,t1}$, $M_{Rd,t2}$, $M_{Rd,b1}$ and $M_{Rd,b2}$ are calculated for the actual reinforcement at the respective end of the span as described previously. $\sum M_{Rc,i}/\sum M_{Rb,i}$ is the ratio of sums for bending capacities of columns and beams connected to the current joint. If $\sum M_{Rc,i} > \sum M_{Rb,i}$, ratio is assumed to be equal to 1.0.

After the capacity shear forces are calculated, you can include them into the reinforcement design. Click again "**Reinforcement**" and "**Drawing**" to obtain the final results.

Deflections and crack widths

You can use Beam Expert to calculate elastic and inelastic deformations and crack widths of beams. SLS load combination is used for these calculations.

Elastic deflections

Click on the "**Deflections**" button immediately after the analysis to see a diagram of elastic deflections. They are determined using mean elastic modulus of concrete and gross area properties of concrete section without reinforcement.

Inelastic deflections

In order to see the inelastic deflections, go to „**Reinforcement**" and „**Drawing**" pages first by clicking the respective buttons and then go back to „ **Deflections**" . Calculations are performed for SLS including nonlinear concrete characteristics such as creep, shrinkage, cracks and actual reinforcement. For each section, crack opening check is performed and crack widths are calculated if necessary.

For calculation of inelastic deflections, each span is divided into 12 elements. Top and bottom reinforcement is determined for each element. Crack opening bending moment is calculated. The element is classified as

cracked if design bending moment is greater than crack opening moment or as non-cracked otherwise. Nonlinear stiffness is calculated for cracked or non-cracked element. Then analysis is restarted and deflections are calculated as a result.

Reinforcement design

Beam Expert performs bending and shear design according to Eurocode. The same procedures are used as in the RC Expert module. You can find a detailed description of the design procedures in the RC Expert user manual. Calculations are performed for all sections along each span. Required reinforcement area diagrams are obtained for main (top and bottom) and shear reinforcement. You can start the design procedure by clicking the „**Reinforcement** ” button .

Effective depth for bending design is assumed to be $d = h - 5$ cm. Shear design is performed for two leg links. Reinforcement area A_{sw} [cm^2/m] is provided for one leg. Additional tensile force ΔF_t from shear design is calculated in main reinforcement, according to Eurocode 2. It is included by widening the main reinforcement diagram.

Report

You can generate a detailed report in **HTML** format for each task by clicking the "**Results**" button. The report is opened in a web browser (Internet Explorer by default). Most office programs like MS Word can edit **html** files. Report filename is **data_file_name.html**.

It comes together with a folder **data_file_name.html_files**. Always keep the report file and the folder together, otherwise pictures and formatting will be lost.

Detailing and drafting

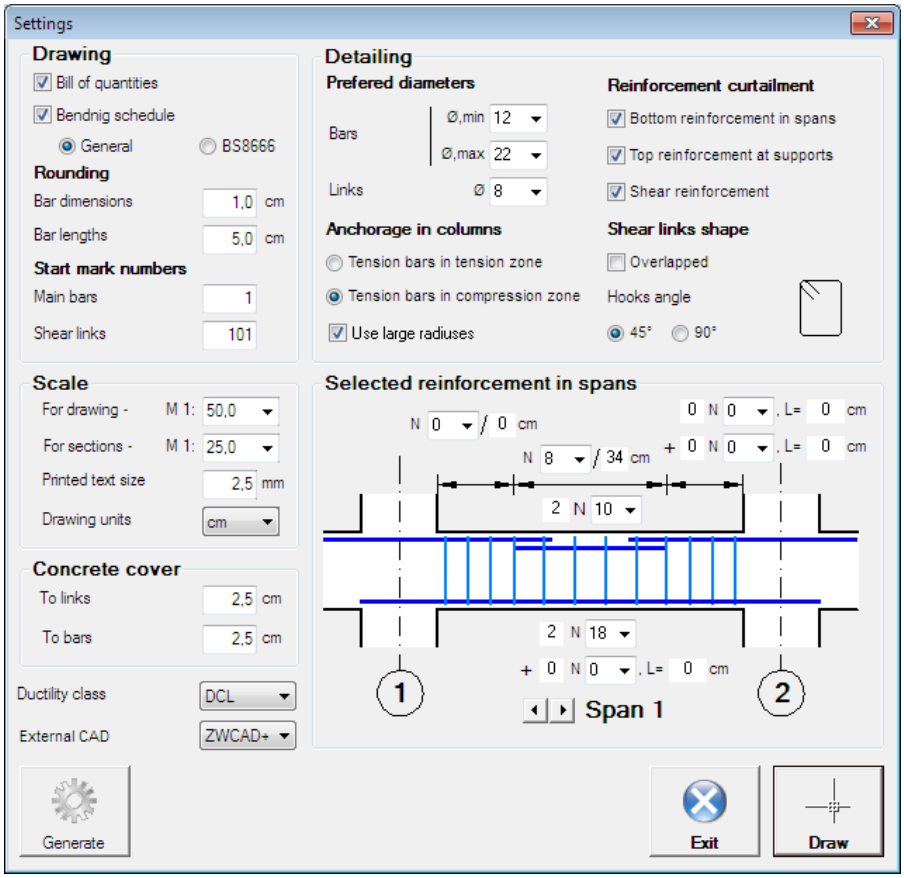
Beam Expert is capable of automated reinforcement detailing. After that you can export ready to use drawings directly to AutoCAD or ZWCAD. Unlike purely detailing software where you have to enter count, diameter, shape and dimensions for each bar or shear link, Beam Expert can generate all of the above data automatically, based on the required reinforcement diagrams. Eurocode detailing requirements are automatically included, such as minimum and maximum reinforcement ratios, bar and link spacing, anchoring and splicing, bending radiuses. Eurocode 8 requirements for seismic design are also included. That allows you to generate detailed reinforcement drawings that correspond to code requirements with minimum input data.

You can start reinforcement detailing and drawing by clicking the „**Detailing**” button on the main toolbar. The “**Settings**” dialog appears on screen where you can do the following:

- Select options for reinforcement detailing and drawing layout;
- Automatically generate reinforcement bars and shear links by covering the required reinforcement diagrams;
- Modify counts and diameters for top, bottom and shear reinforcement in each span;
- Generate the drawing with the selected reinforcement.

Settings

The settings dialog is displayed on the picture below. Detailed description of all available options is provided further in this chapter.



Drawing

Bill of materials

Bill of materials (**BOM**) includes weight of reinforcement (kg), total and by bar size, as well as concrete volume (m^3) and formwork area (m^2).

Bending schedule

Bending schedule includes information about diameter, length, count, shape and dimensions for each bar mark. You can select between two styles of scheduling: “Standard” and „BS8666”. The standard style includes drawings with dimensions for each bar mark. BS8666 style is according to British Standard BS8666:2005. Each bar shape is represented by shape code and all dimensions (A, B, C etc.) are filled in a table. Bars are not drawn except for shape code 99.

Bending schedules and BOM include only the current beam. If you are going to have several beams in a single drawing and you want to make a common schedule and BOM for all of them, you can do the following: Switch the scheduling and BOM options off in Beam Expert. Export the drawings to AutoCAD or ZWCAD+. You can select starting bar mark number for each beam to continue from the previous one. Use the scheduling command from RC Plug-in module to generate schedule and BOM inside AutoCAD or ZWCAD+.

Rounding

You can specify rounding steps for bar dimensions and total bar lengths. They are 5 mm and 25 mm by default. Links are not affected by this options. They are always rounded to an integer number.

Numbering

Enter starting mark numbers for shear links and bars separately. You can continue the numbering from the previous beam.

Scale

You can set different scales for elevations and sections. Specify text size in millimeters as it should appear on the printouts. Actual text size on the screen is automatically calculated according to the scale. You can also select different drawing units (mm, cm or m).

Concrete cover

Concrete cover is defined as the distance between concrete surface and surface of shear links and bars, respectively. Actual distance from bars to concrete edge is taken as the greater value of:

- cover to main bars or
- cover to shear links + link diameter.

External CAD

You can export the drawing to different CAD systems. You have to select the preferred system (ZWCAD+ or AutoCAD) in the combo box. See “Export to AutoCAD and ZWCAD” further in this manual.

Ductility class

Detailing requirements for different ductility classes are included. The program automatically applies the requirements for the selected ductility class during the detailing process. You should select one of the following possible values from the combo box: DCL, DCM or DCH. For more information, see "**Detailing requirements to Eurocode 2 and Eurocode 8**" bellow. If you need to design to Eurocode 2 only, select DCL. Otherwise, Eurocode 8 is applied.

Detailing

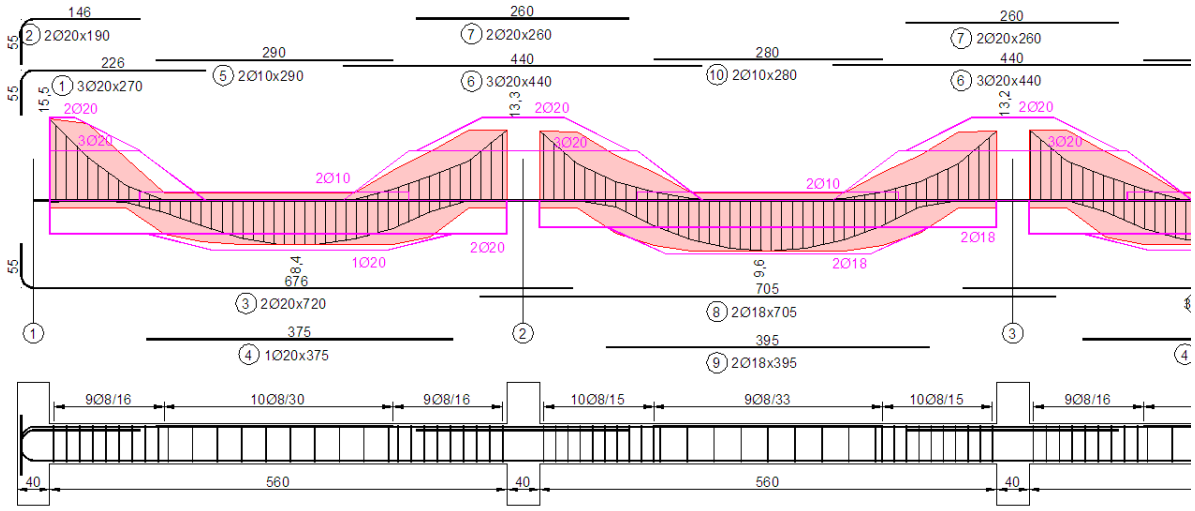
The program automatically selects counts and diameters for bars and shear links based on the required reinforcement diagrams obtained by the design. This is performed after the settings dialog is opened for the first time or when you click the “**Generate**” button.

You can set limits for diameters which can be used by the program by setting the “Preferred diameters” option. For main reinforcement select minimum and maximum diameters [$\mathbf{Ø_{min}}$; $\mathbf{Ø_{max}}$]. Lower values of $\mathbf{Ø_{max}}$ will increase the number of bars. For shear links you can select one preferred diameter **Ø**. It is used for the entire beam, if possible. The program will use larger diameter only if link spacing gets lower than the minimum.

You can select whether to use curtailment separately for bottom, top or shear reinforcement. Without curtailment, all bars in the span (or above the support) have equal lengths and count is calculated for the maximum required area. Link spacing is calculated also for the maximum shear reinforcement area and it is uniform along the whole span. The standard length for top bars above supports (without anchoring) is ¼ of

the longer adjacent span. If top reinforcement diagram goes outside this zone, it is covered by the top reinforcement in spans. For cantilever spans, top bars above the support are projected to the free end of the cantilever.

When the curtailment option is selected, part of reinforcement bars (≥ 50%) is extended at full length and the remaining part is curtailed within the zone that is not covered by the capacity of the first part. Shear links are divided into 2 zones – with denser spacing within end quarters of the span and sparser in the middle half. With curtailment, quantity of steel gets lower, but the number of different bar marks increases and detailing becomes more complicated. It is appropriate for larger and more heavily loaded beams. Example of reinforcement curtailment with Beam Expert is shown on the picture below:



Detailing requirements to Eurocode 2 and Eurocode 8

All parameters, used for automated detailing of beams are listed in the table below. References to the corresponding sections of Eurocode are provided in brackets.

Longitudinal reinforcement		**DCL Non-seismic element EC2**	**DCM Seismic element EC8**	**DCH Seismic element EC8**
Minimum diameter	$d_{bL,min}$	10 mm	12 mm	14 mm (**5.5.3.1.3 (5)**)
Minimum tensile reinforcement ratio	ρ_{min}	$0.26 \cdot f_{ctm}/f_{yk}$ 0.0013 (**9.2.1.1 (1)** and **NA.2.74**)	$f_{ctm}/2f_{yk}$ (**5.4.3.1.2 (5)**)	$f_{ctm}/2f_{yk}$ (**5.5.3.1.3 (5) a**)
Minimum top reinforcement in spans		-	-	2N14, (**5.5.3.1.3 (5) b**) $0.25A'_{s,cr}$ (**5.5.3.1.3 (5) c**)
Maximum reinforcement ratio	ρ_{max}	0.04 (EN 1992-1-1, **9.2.1.1 (3)** и **NA.2.75**)		
Maximum ratio for tensile reinforcement in critical zones	$\rho_{cr,max}$	-	$\rho'_{cr} + 0.0018\, f_{cd}/(\mu_\phi\, \varepsilon_{sy,d}\, f_{yd})$ (**5.4.3.1.2 (4)** и **5.5.3.1.3 (4)**)	

Minimum clear spacing between bars	a_{min}	max{$d_{bL,max}$, d_g + 5 mm, 20 mm} [1)] (EN 1992-1-1, **8.2 (2)** и **NA.2.70**)	
Maximum spacing to centers of bars	a_{max}	250 mm	
Mandrel diameter for bending	d_m	For Ø ≤ 16 mm - d_m = 4Ø For Ø > 16 mm - d_m = 7Ø	
Anchorage length	l_{bd}	$f_{bd} = 2.25\eta_1\eta_2 f_{ctd}$, $l_{b,rqd} = d_L/4 \cdot \sigma_{sd}/f_{bd}$ $l_{bd} = \alpha_1\alpha_2\alpha_3\alpha_4\alpha_5 l_{b,rqd} > l_{b.min}$ $l_{b.min}$ = max{0.3$l_{b,rqd}$,10d_L, 10 cm}	$l_{bd} + 5d_{bL}$ (**5.6.1 (3)**)
Extension of bars in the next span after an internal joint		-	l_{cr} (**5.6.2.2 (4)**)
Maximum ratio of bar diameter to the size of the column section	$\frac{d_{bL}}{h_c}$	-	$7.5 f_{ctm}/\gamma_{Rd} f_{yd} \cdot (1 + 0.8\nu_d)$ – at end joints [2)] $7.5 f_{ctm}/\gamma_{Rd} f_{yd} \cdot (1 + 0.8\nu_d)/(1 + 0.75 k_D \rho'/\rho_{max})$ – at internal joints (**5.6.2.2 (2)**)

[1)] Gravel size is asuumed to be d_g = 20 mm by default.

[2)] Normalized axial load is assumed to have a minimum value of ν_d = 0.1, conservatively. There is no data for internail forces in connecting elements when using Beam Expert standalone.

Shear reinforcement		**DCL**	**DCM**	**DCH**
Minimum reinforcement	$d_{w,min}$	6 mm (**5.4.3.1.2 (6) a** and **5.5.3.1.3 (6)**)		
Minimum spacing between shear reinforcement centers	s_{min}	50 mm		
Maximum spacing between shear reinforcement centers	s_{max}	0.75d (1 + cotα) (EN 1992-1-1 **9.2.2 (6)** и **NA.2.80**)		
Maximum spacing between shear reinforcement centers in critical zones	$s_{cr,max}$	-	$h_w/4$ 24d_w 8d_{bL} 225 mm (**5.4.3.1.2 (1) b**)	$h_w/4$ 24d_w 6d_{bL} 175 mm (**5.5.3.1.3 (6)**)
Distance from first shear link to column edge		50 mm (**5.4.3.1.2 (1) c**)		
Anchoring length inside the concrete	l_{bw}	10d_w (**5.6.1 (2)**)		
Critical zone length	l_{cr}	$l_n/4$ zone of condensation [1)]	$l_n/4$ [2)] h_w (**5.4.3.1.2 (1)**)	$l_n/4$ [2)] 1.5h_w (**5.5.3.1.3 (1)**)

[1)] If you select the reinforcement curtailment option, link spacing is condensed at distance of ¼ of span length near the supports.

[2)] In order to simplify the detailing, the condensation zone and the critical zone are merged. Most unfavorable parameters are used from both zones.

List of symbols:

f_{ctm} – mean concrete tensile strength;

f_{yk} – characteristic reinforcement yield strength;

$A'_{s,cr}$ – compressive reinforcement area in critical zone;

ρ'_{cr} – compressive reinforcement ratio in critical zone;

f_{cd} – design concrete compressive strength;

μ_ϕ – minimum curvature ductility factor in critical zone $\mu_\phi = 1 + 2(q - 1)T_c/T_1$;

- q – behavior factor, Table 5.1.
  It is assumed conservatively that q = 3.3 for DCM and q = 4.95 for DCH;
- T_c – vibration period at the end of the horizontal part of the response spectrum;
- T_1 – vibration period for first mode shape. It is assumed: T_c/T_1 = 1.0, which is conservative for greater values of T_1;

$\varepsilon_{sy,d}$ – design value for reinforcement yield strain = f_{yd}/E_s

f_{yd} – design reinforcement yield strength;

$d_{bL,max}$ – maximum longitudinal bar diameter;

d_g – gravel grain size;

γ_{Rd} – safety factor equal to 1.0 for DCM and 1.2 for DCH;

ν_d – normalized column axial load $\nu_d = N_{ed}/f_{cd}A_c$;

k_D – factor equal to 2/3 for DCM and 1.0 for DCH;

ρ' – ratio of the compressive reinforcement anchored into the column;

ρ_{max} – maximum tensile reinforcement ratio $\rho_{max} = \rho' + 0.0018\, f_{cd}/(\mu_\phi\, \varepsilon_{sy,d}\, f_{yd})$;

d – effective section height;

α – angle between shear reinforcement and beam axis;

h_w – beam section height;

d_w – shear reinforcement diameter;

d_{bL} – longitudinal reinforcement diameter (minimum);

l_n – span clear length.

Selected reinforcement in spans

The program automatically selects counts and diameters for all bars and shear links based on the design results. The procedure is started with the "**Generate**" button. You can review and edit the reinforcement for each span separately. You can browse among spans using the  buttons. In order to modify the top reinforcement at support 1, go to span 1 and click the left arrow. If you press the "**Generate**" button once again, it will discard all changes you have made and will regenerate the whole reinforcement.

Drafting

When you press the "**Draw**" button, the entire drawing is generated and the "Settings" dialog is closed. If you go back to the "Reinforcement" page, you will see diagrams for theoretical reinforcement area vs. actual reinforcement area.

Working with Design Expert graphical environment

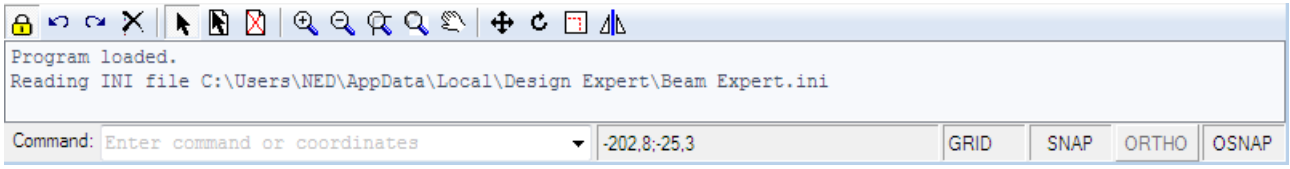
All drawings are generated in the internal Design Expert graphical environment first. There you can view, modify and align objects before exporting them to ZWCAD+ or AutoCAD. The graphical environment includes a basic set of commands for drawing and editing.

Commands

How to enter commands?

You can use several ways to enter a command in this program:

- Type it into the command line;
- Type the short version (command alias);
- Press a button on the toolbar;



Alternatively, instead of typing you can select the command from a drop down list by clicking the small arrow right to the command line. Some commands may require you to select objects or enter coordinates. You should watch the prompt on the left side of the command line. Press enter or right mouse button to complete a command that is running. You can cancel a command prematurely by pressing Esc or right mouse button. Commands generate various output including error or warning messages, results and general information intended for the user. You can find it in the output window just above the command line. You can start the previous command by pressing Enter or Space key instead of typing it again or pressing a button.

List of commands

A list of all available commands including icons, aliases and short descriptions is provided in the table below. You can find detailed descriptions of all commands further in this manual.

	Command	Alias	Description
	ZWCAD+ AUTOCAD	CAD	Export the current drawing to AutoCAD/ZWCAD+.
	COPY	CP, CO	Replicate the selected objects by moving, rotating, mirroring or scaling.
	COPYBITMAP	CB, COPYBMP	Copy the current drawing as Bitmap to system clipboard where it is available to paste in other programs.
	COPYMETAFILE	CM, COPYWMF	Copy the current drawing as Metafile.
	DELETE	E, D, DEL, ERASE	Delete selected objects from both screen and memory.

	DESELECTALL	DE, DESEL, DESELECT	Deselect all objects.
	DISTANCE	DI, DIST	Measure distance and angle between points.
	EXIT	QUIT	Close the program and exit.
	GRID	GR	Turn grid on and off.
	HELP		Display user manual.
	MIRROR	MI	Mirror the selected objects about a line defined by two points.
	MOVE	M, MO	Move the selected objects along a vector defined by two points.
	NEW	N	Create a new file.
	OPEN	O	Open an existing file from the disk.
	ORTHO	OR	Turn orthogonal drafting mode on and off.
	OSNAP	OS	Turn object snap mode on and off.
	PRINT	PR, PRN	Send the current drawing to the printer.
	REDO	RE	Restore the last command after UNDO.
	REDRAW	RD	Redraw the screen view.
	ROTATE	RO	Rotate the selected objects about a specified center point and angle.
	RTPAN	PA, PAN	Move the screen view to other part of the drawing.
	SAVE	S	Save the current data to a file on the disk.
	SCALE	SC	Scale the selected object with specified center point and scale factor.
	SCRIPT		Save a script file (*.scr) with all AutoCAD commands ncessary to create the current drawing in AutoCAD.
	SELECTALL	A, ALL, SELALL	Select all objects in the drawing that are not hidden or locked.
	SNAP	SN	Turn snap-to-grid mode on and off.
	UNDO	U	Undo the last command.
	ZOOMIN	ZI, Z+	Zoom in the screen view by factor of 1.5.
	ZOOMLIMITS	ZL, ZA, ZE	Zoom the screen view in order to fit all objects inside the program window.
	ZOOMOUT	ZO, Z-	Zoom out the screen view by factor of 0.5.
	ZOOMWINDOW	ZW	Zoom the screen view in order to fit inside the specified rectangle.

Undo wrong action or command

Click the button or type the UNDO command.

It cancels the results from the last command and recovers the previous drawing state. You can undo only one step back. If you need to go back further, use the other commands to recover the original drawing state.

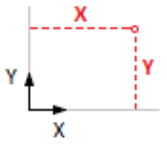
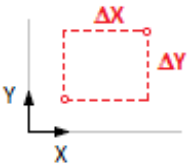
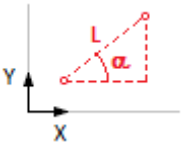
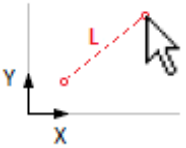
Redo a command that has been undone

Click the button or type the REDO command.

It repeats the last command in case it has been accidently undone. REDO must follow the UNDO command immediately before any other command. Otherwise, the command cannot be recovered.

Points and coordinates input

Design Expert has its own CAD environment where you can create and modify drawings. Some commands require the user to enter coordinates of points. You can do this by clicking with the mouse in the drawing window or by typing the coordinates in the command line. Typing input should follow some standard formats as described below. Coordinates can be absolute or relative to the previous point.

Type	Input format	Example	Description	Picture
Absolute	$X;Y$	10,5;15	Values are defined in global coordinate system Oxy.	
Relative	**_**$\Delta X;\Delta Y$ **@**$\Delta X;\Delta Y$	@25;35	Relative distances "25" и "35" to the previous point along X and Y, respectively.	
Polar	<$\alpha^o;L$	<45;100	Distance of "100" is measured to the previous point at 45° angle from X axis.	
Distance	L	50	Distance of "50" to the previous point measured towards mouse cursor.	

Press Enter or Space after you enter the coordinates in the command line. If you want to enter points with the mouse, you have to move the cursor to the required location and click with the left mouse button. You can see the current coordinates of the cursor in the status bar located at the bottom of the main window. You can use several precision tools that can help you to get the exact coordinates when clicking:

- **GRID** – shows a uniform grid of dots over the working area of the drawing;
- **SNAP** – rounds the coordinates to a specified step along X and Y;
- **ORTHO** – orthogonal drawing mode. Current point is aligned to horizontal or vertical line with the previous point depending on the mouse position;
- **OSNAP** – gets the coordinates of an existing point in the drawing, when you move the mouse or click over it closer than a specified range. If several points are located within the range, the closest one is returned. When a point is snapped, an "×" mark appears on the screen. It is always the same symbol regardless the point type.

You can switch on and off the precision tools using the respective buttons on the status bar or by typing the respective commands in the command line.

Manage the screen view

The drawing is located in the model space and it is defined in global coordinate system Oxy. Then it is projected to the screen to certain scale. You can see only a part of the model space that is visible within the program window. We will call this “screen view”. You can scale and move the screen view over the drawing using ZOOM and PAN commands. That is how you can work with different parts of the drawing as necessary.

Zoom in and out

If you have a wheel mouse, you can zoom in and out by rotating the wheel forward and backward. The center of the transformation (the point that does not move) is assumed to be the current position of the cursor. You can move quickly to different parts of the drawing by positioning the cursor at different locations and zooming in and out. Also, you can use some additional commands as follows:

ZOOM IN – zooms in the screen view with one step;

ZOOM LIMITS – zooms the screen view so that all visible objects fit inside the screen;

ZOOM WINDOW – zooms the screen view in a user defined window. When you start the command, you have to enter two points, at the opposite corners of the window;

ZOOM OUT – zooms the screen view out with one step.

Pan

You can move the screen view at preferred direction in order to see other parts of the drawing. If you have a three-button mouse, you can use the middle button to pan. Press and hold the middle button, drag it to the new location and release the button. When you press the button, the cursor changes to and when you release it, the old cursor is restored back.

Alternatively, you can use the *RTPAN* command. It requires two points to define the length and the direction of movement (towards the second point). Since *RTPAN* is a command like any other, you have to finish the previous command before that. Unlike *RTPAN*, the middle button method can be used transparently inside any command without interrupting it.

Copy screen

You can copy the screen view to the clipboard any time and insert it into other programs using Paste command or *Ctrl+V.* You can use the following commands for coping:

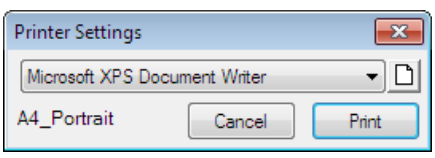
COPYBITMAP – copies the screen image as Bitmap;

COPYMETAFILE – copies the screen image as Metafile.

Bitmap is a raster format file that stores information about colors of separate pixels. Metafile is a vector format file that stores coordinates of graphical objects. The boundaries of the copied image match the boundaries of the program window. Only objects that are visible on the screen will appear in the image. For best results, you can stretch the program window beforehand in order to fit the drawing tightly in the window without white spaces.

Print screen

You can send the screen view directly to the printer by pressing the button or typing the *PRINT* command. A setup dialog appears on screen. Select the required printer device from the list. You can change paper size and orientation as well as other options by clicking the button. Press the “Print” button to finish.



Modify objects

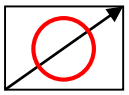
Block editing mode vs free mode

Graphics in Design Expert are represented by basic objects like lines, polylines, circles, texts, dimensions etc. They are grouped in blocks in order to form more complex objects like reinforcement bars, sections or entire elements. Each block is attached to one or more grips that are displayed as small blue boxes. By default, the drawing is locked and you can move only entire blocks using the respective grips. This is called “block mode”. You cannot modify separate objects within blocks. If you want to do that, you have to unlock the drawing first. Locking and unlocking is performed by clicking the respective buttons .

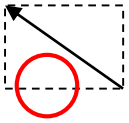
Select

Selection is a way to determine which objects should be affected by a certain command. You can select objects either before or after the command. There are several ways to select objects:

- Single – click on the object outline with the left mouse button. The outline should intersect the cursor selection box . If there are no object at the specified point, the program automatically continues to window selection mode.



- Window – you have to enter two points at the opposite corners of a window. If you draw the window from left to right, all objects that fit entirely inside are selected. If you draw the window from right to left, all objects that intersect or fit inside the window are selected. The window is displayed with solid line in the first case and dashed line in the second.



- All – selects all visible and unlocked objects. Click the button or type *SELECTALL* to start the command.

Selected objects are redrawn in red.

Deselect

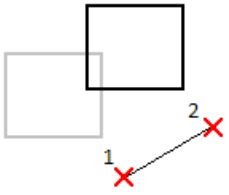
Deselection is performed in the same way as selection but additionally you should hold the Shift button. Alternatively, you can click an object with the right mouse button. In order to deselect all objects, press Esc, click the button or type *DESELECTALL*.

Delete

Click the button or type *DELETE*. All selected objects are erased both from screen and memory.

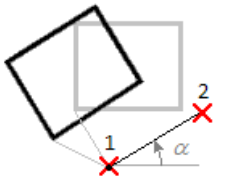
Move

Moves the selected objects along a vector defined by two points. Click the button or type *MOVE*. Then enter first and second point and press Enter or click the right mouse button.



Rotate

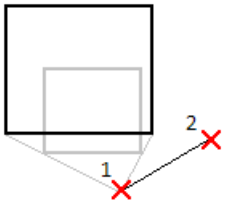
Rotates the selected objects around a center and with angle defined by user. Click the button or type *ROTATE*. Then enter first and second point and press Enter or click the right mouse button. The first point defines the center of rotation and the second is for the angle. The angle is measured between the line and the +X axis counterclockwise. You can also enter the exact value of the angle using polar



coordinate input format. Type "<α;1" in the command line instead of clicking the second point, where α should be the rotation angle in degrees.

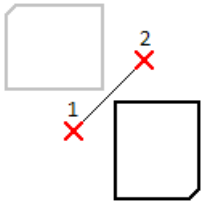
Scale

Scales the selected objects with a center and scale factor defined by user. Click the button or type *SCALE*. Then enter first and second point and press Enter or click the right mouse button. The first point represents the center of transformation. Scale factor is defined as the distance between the first and the second point. Alternatively, you can type the scale factor in the command line instead of entering a second point.



Mirror

Mirrors the selected objects about a line defined by user. Click the button or type *MIRROR*. Then enter first and second point and press Enter or click the right mouse button.



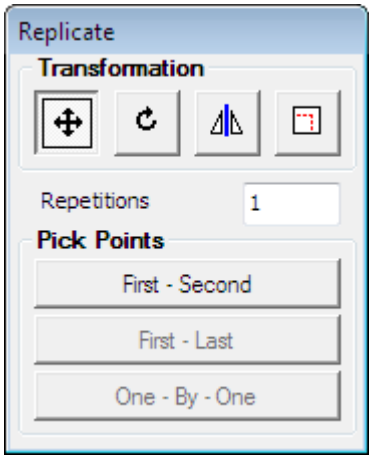
Stretch

When the drawing is unlocked, you can stretch separate objects like points, lines, polylines, dimensions, circles, polygons and texts by “dragging” with the mouse. Select the object and click on a point (end, middle or center point) to “catch” it. Then move the cursor to a new location and second click to “release” it. Texts are selected and moved using their base points displayed as small circles. If you stretch a line, polyline or polygon and you hold shift before the second click you will insert a new vertex.

When the drawing is locked then you work in block editing mode. You can move entire blocks by stretching the respective grips. First, you have to select a grip by clicking with the mouse. Then, click again on the grip to “catch” it, move it to the new location and click to “release” it.

Copy

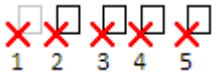
Creates one or multiple copies of the selected objects using one of the available transformations (move, rotate, scale or mirror). Click the button or type the *COPY* command. Select objects and press Enter or click the right mouse button. A settings dialog appears on screen. Select method of transformation using the icons on the top, number of repetitions and method of pointing:



- **First – second** – click two points that define the distance between two consecutive objects;

- **First – last** – click two points that define the distance between the first and the last object. All other objects will be distributed evenly between them;

- **One - by - one** – click a base point first. Then you have to enter separate points to define the location of each object independently.



Coping is not available for some objects in some modules.

Export to ZWCAD+ or AutoCAD

You can export the drawing directly to ZWCAD+ or AutoCAD by clicking the  /  button from the main toolbar. It is always the same button, but the icon is different depending on the selected “External CAD” option in the settings dialog. If you click the arrow ▾ next to the button you can select other CAD system from the drop down menu. Supported versions are ZWCAD+ 2012 to 2015 and AutoCAD 2004 to 2015. If there is an instance of ZWCAD+ or AutoCAD already running, the drawing is sent to the active document. Otherwise, a new session is opened. Alternatively, you can type one of the following commands: *ZWCAD+*, *AutoCAD* or just *CAD*.

The drawing is exported as simple polylines, texts, dimensions, lines, circles and hatches. There are no blocks or any other complex objects, so it is easy to be modified with the standard AutoCAD commands. Current text and dimension styles are used. If you use templates, the drawing will look as any of your other drawings. For best results you have to define "Text Placement" to be "Over the Dimension Line, Without a Leader" in the dimension style settings. Objects are distributed in separate layers. If the required layers do not exist, they are created automatically. Reinforcement output is compatible to Design Expert Plug-in module. You can use it to additionally modify and schedule the reinforcement bars.

For versions not supported by the direct output, you can create AutoCAD command script files. Click the arrow next to the button and select “Save script file *.scr”. Enter file path and name and click “Save”. Then you can load the saved script into ZWCAD+ and AutoCAD using the *SCRIPT* command or menu "Tools\Run Script...“.